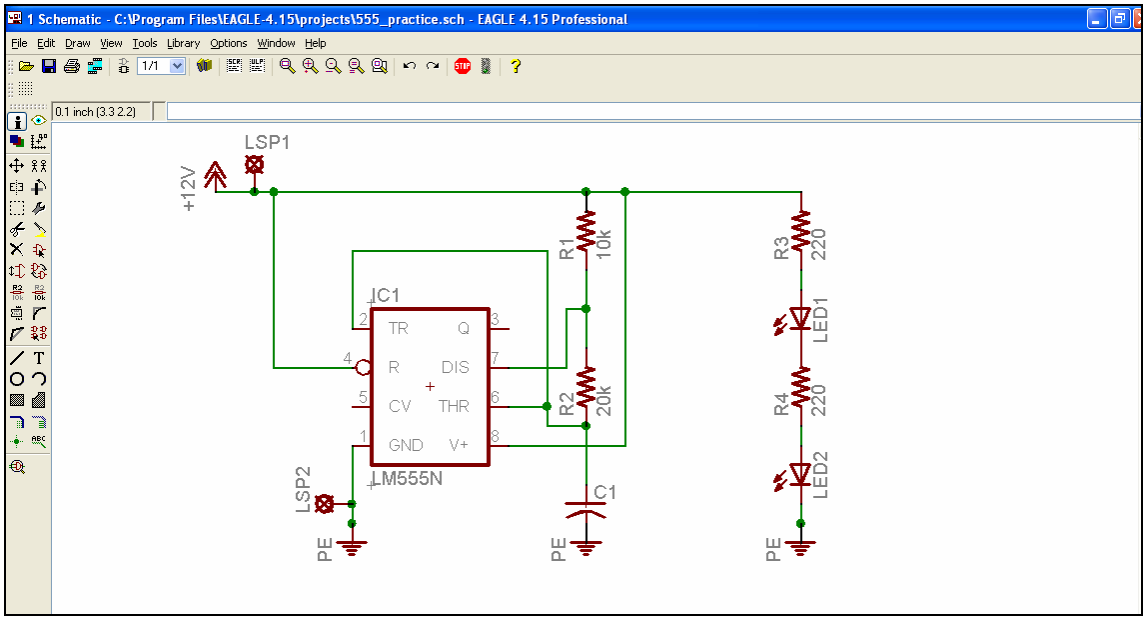
PCB Design and Fabrication
Quick Start Guide

This tutorial describes the processes involved in producing a single layer PCB.

Note: Any Eagle CAD instructions listed within the tutorial are executable using tool bar script box by entering the name of the command icon and pressing enter**.**

Creating a Board From a Schematic

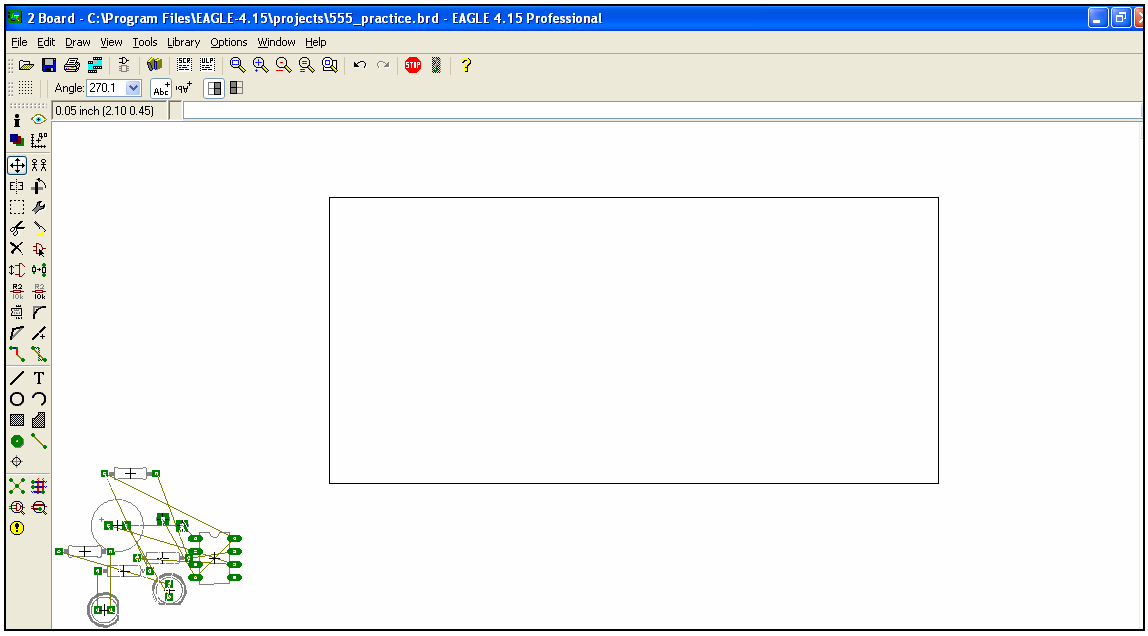
1. Create a schematic using Eagle Schematic.



2. Click on the “Board” icon when the schematic is complete. This opens up Layout Editor and creates a board file with the same name as that of the schematic.

Creating a Board Layout

In the screen shot below, the circuit components and “ratsnest” are on the left side of the window. Eagle uses the term, airwires, to describe the rubber band-like connections between components. The black frame represents the board outlines.

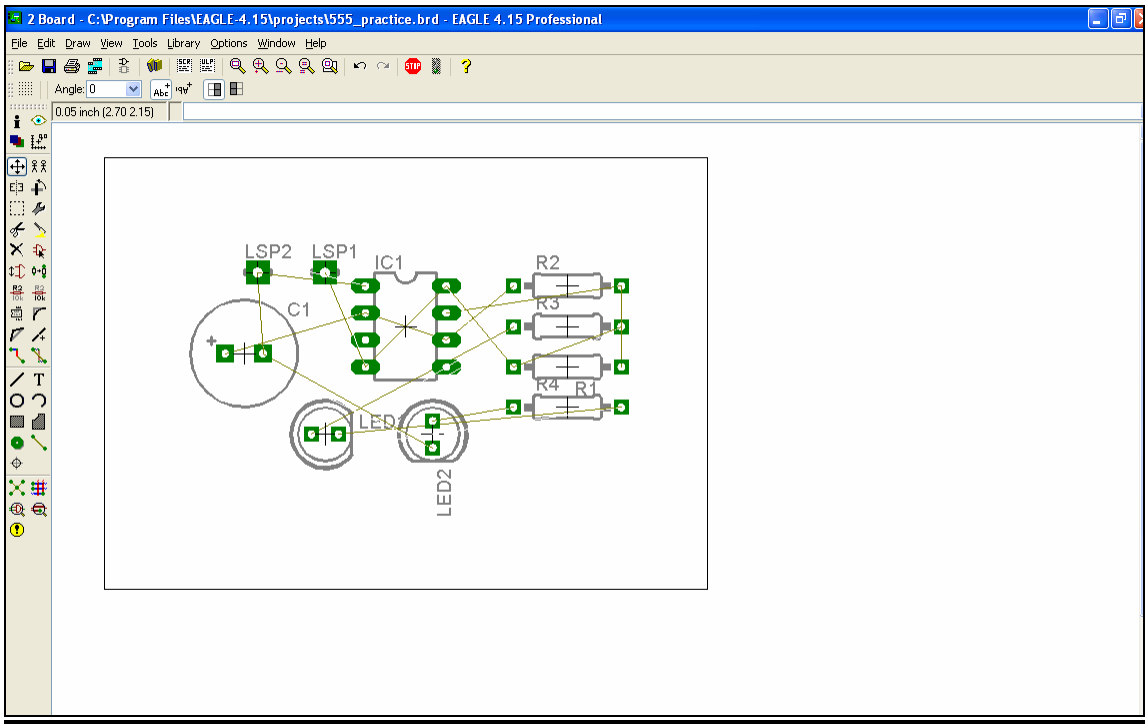

2 Board - C:\Program Files\EAGLE-4.15\projects\555_practice.brd - EAGLE 4.15 Professional
File Edit Draw View Tools Library Options Window Help
Angle: 270.1
0.05 inch (2.10 0.45)

Create the circuit design within the board area using the Move and Rotate commands.

To display component names:

1. select View>Display/hide layers….or click on the Display command icon
2. click on layer 25 (tNames).

The screen shot below displays the desired circuit layout within the board area.



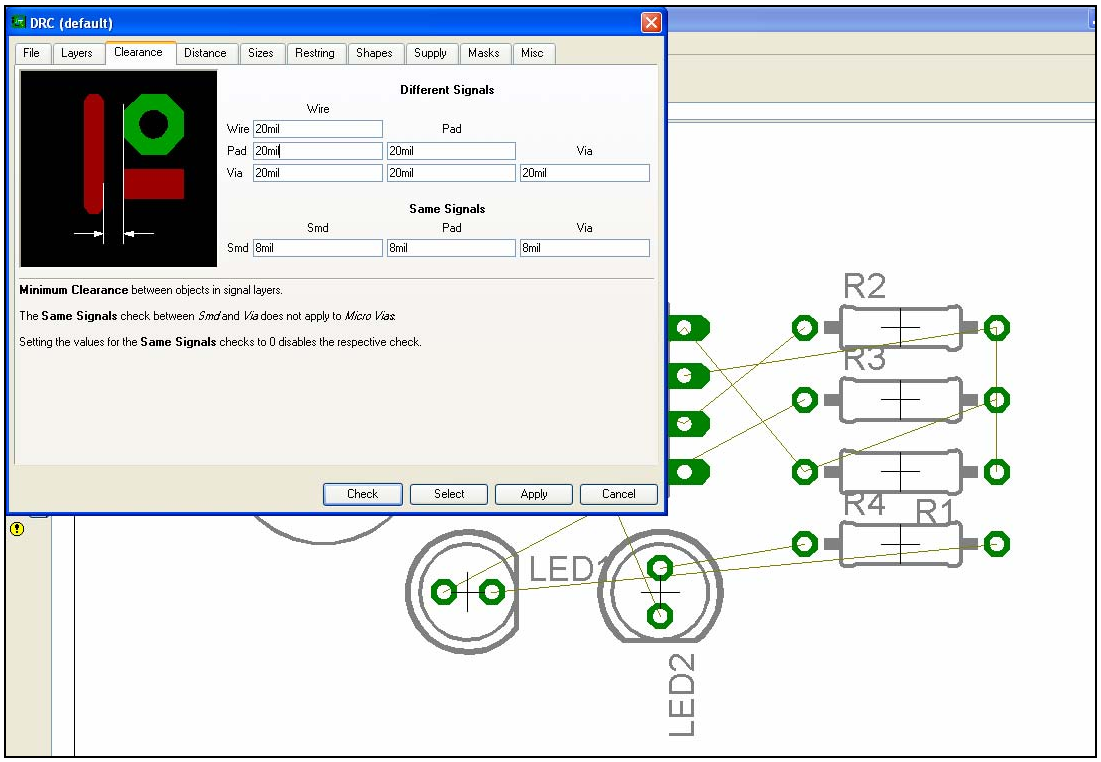
To calculate the shortest possible component airwire connections:

- select Tools>Ratsnest or click on the Ratsnest command icon

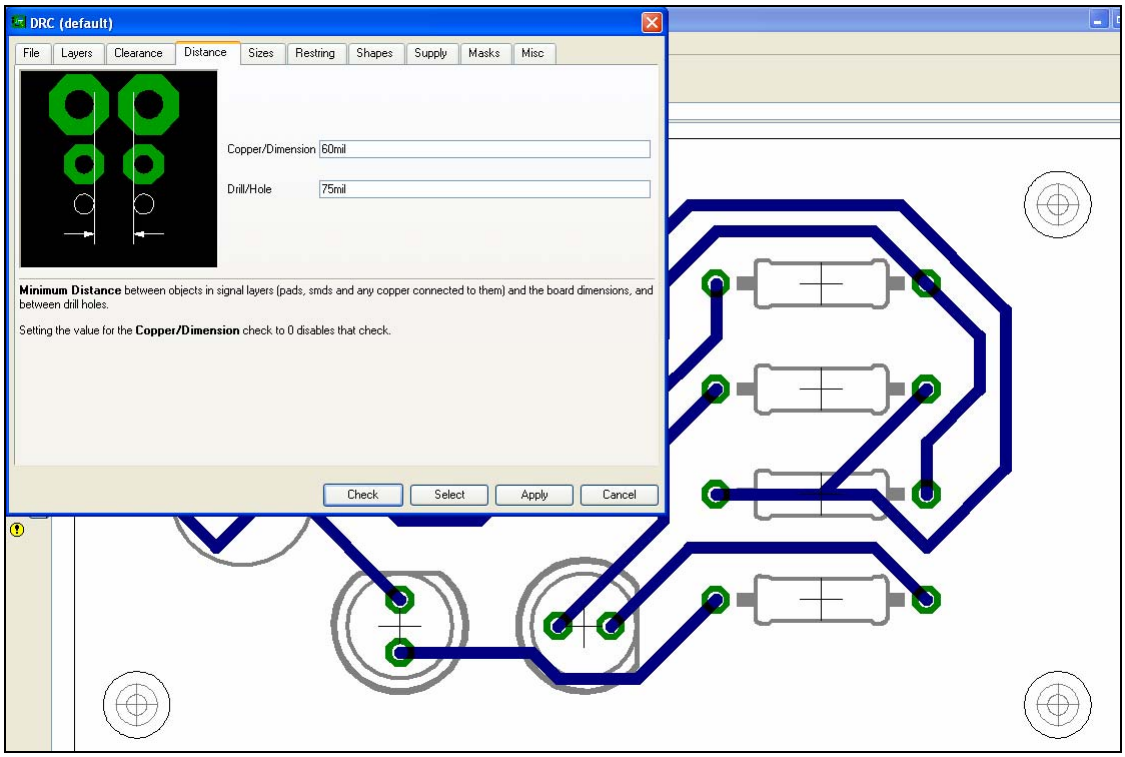
Design Rules

Design Rules specify all parameters relative to board fabrication. To access Design Rules:

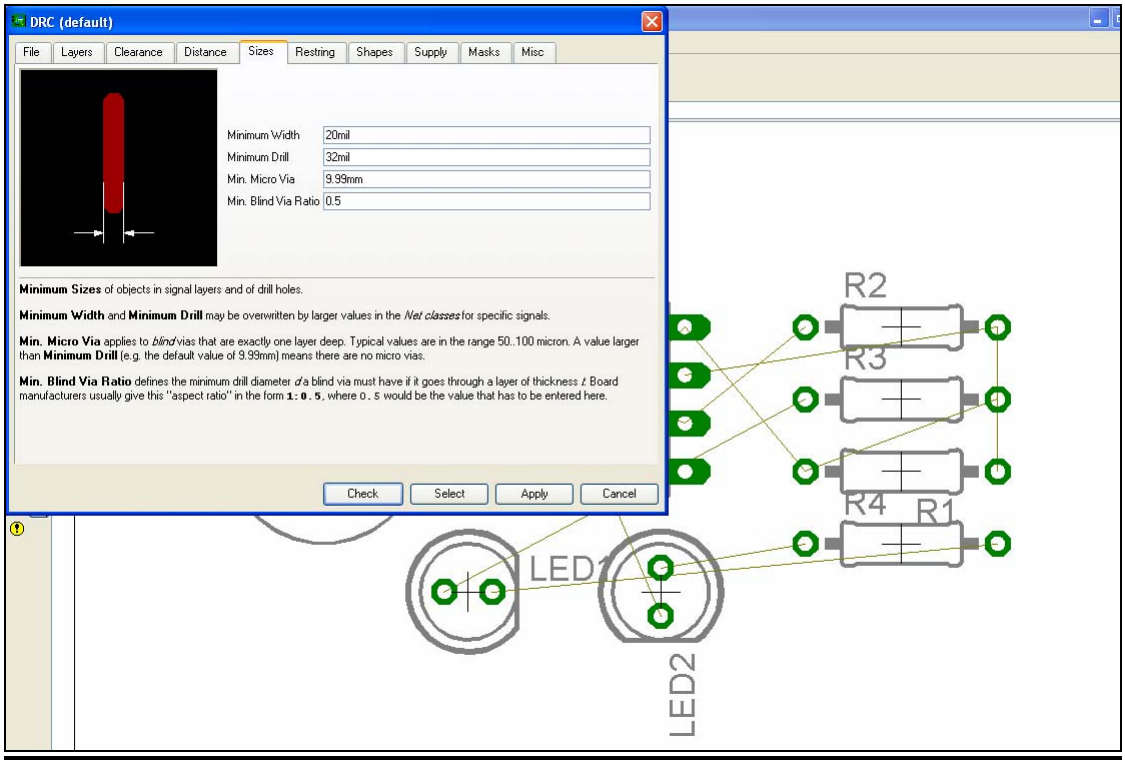
1. Select Tools >Drc (Design rule check) or click on the Drc command icon.
2. Select the Clearance tab.
    a. Set Wire and Pad clearance to 20mil. This value refers to the minimum distance between copper traces, solder pads, or any vias (holes) in the board.



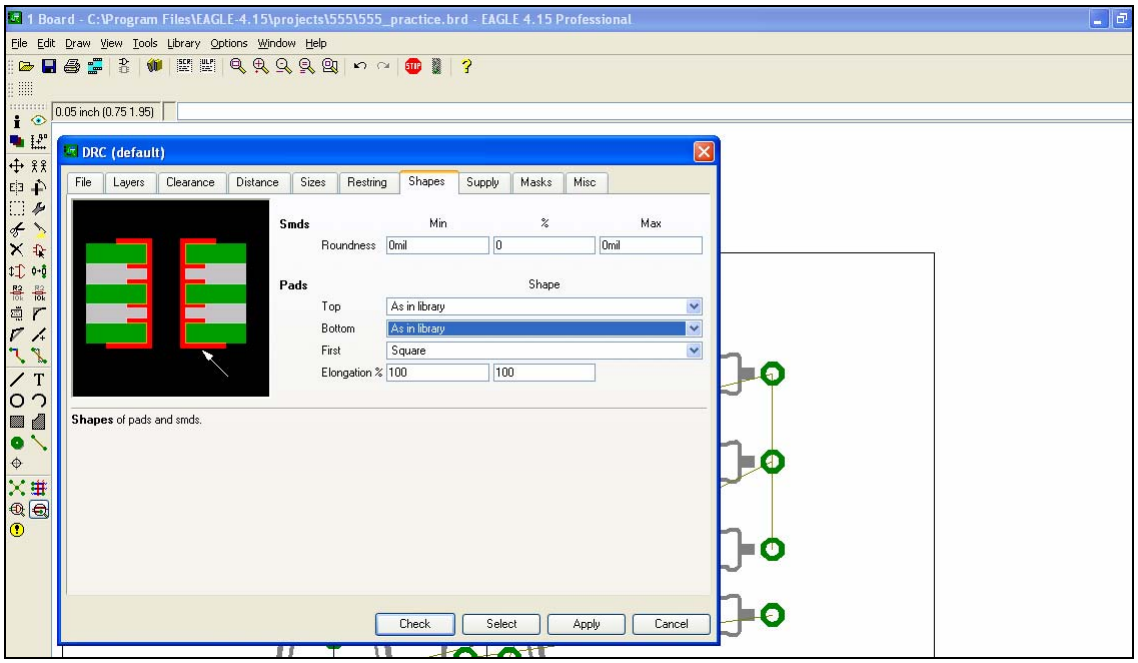
3. Select the Distance tab
    a. 60mil is an adequate distance for setting the Copper/Demension entry.
    b. Set the entry for Drill/Hole to 75mil.



4. Select the Sizes tab
    a. set the minimum width for traces to 20mil
    b. set the minimum drill hole width to 32mil


DRC (default)
File
Layers
Clearance
Distance
Sizes
Restring
Shapes
Supply
Masks
Misc
Minimum Width
20mil
Minimum Drill
32mil
Min. Micro Via
9.99mm
Min. Blind Via Ratio
0.5
Minimum Sizes of objects in signal layers and of drill holes.
Minimum Width and Minimum Drill may be overwritten by larger values in the Net classes for specific signals.
Min. Micro Via applies to blind vias that are exactly one layer deep. Typical values are in the range 50..100 micron. A value larger than Minimum Drill (e.g. the default value of 9.99mm) means there are no micro vias.
Min. Blind Via Ratio defines the minimum drill diameter d a blind via must have if it goes through a layer of thickness L Board manufacturers usually give this "aspect ratio" in the form 1:0.5, where 0.5 would be the value that has to be entered here.
Check
Select
Apply
Cancel
R2
R3
R4
R1
LED1
LED2

5. To change the shape of component pads:
    a. select the Shapes tab. By default, component libraries determine pad shapes.



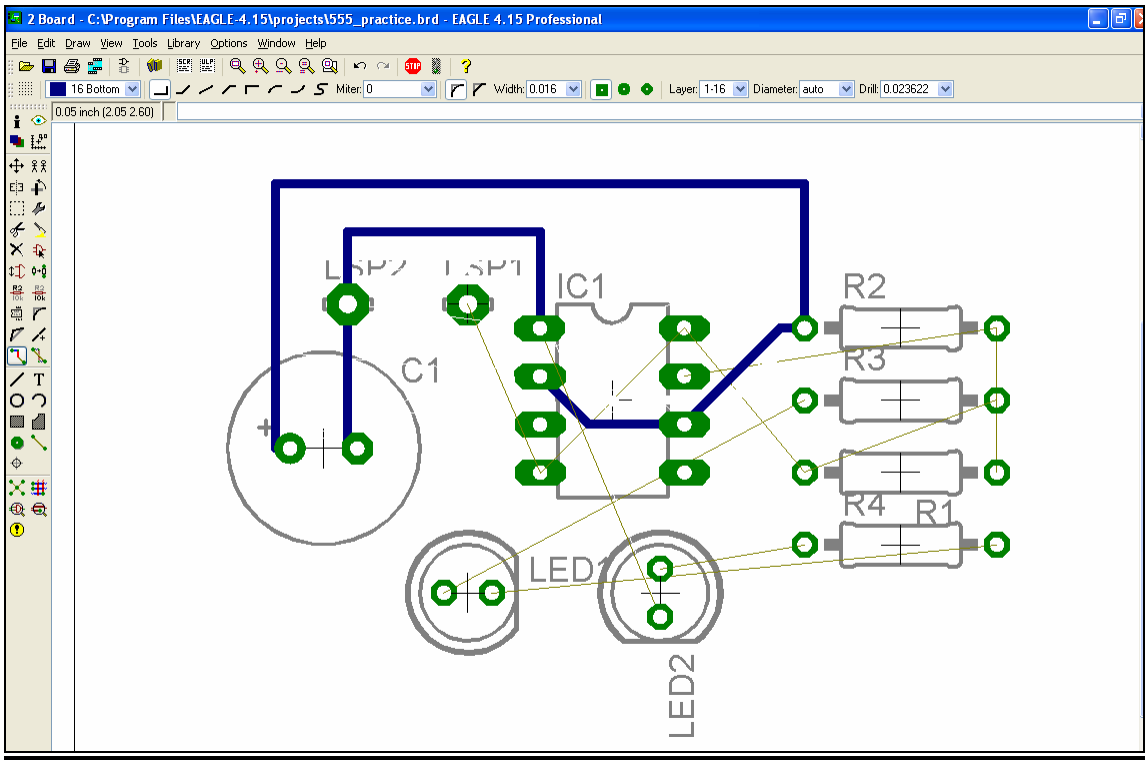
Routing the Board

Placing the Tracks Manually

1. select Edit>Route or click on the Route command icon.
2. click on component pads or airwires and make track connections.

To change track direction or orientation
1. click at the desired point
2. select the appropriate Wire bend icon from the toolbar

The screen shot below shows routing connections from the positive lead of C1 to R2 and pins 2 and 6 of IC1. The negative lead of C1 is routed to LSP2 and to pin 1 of IC1.



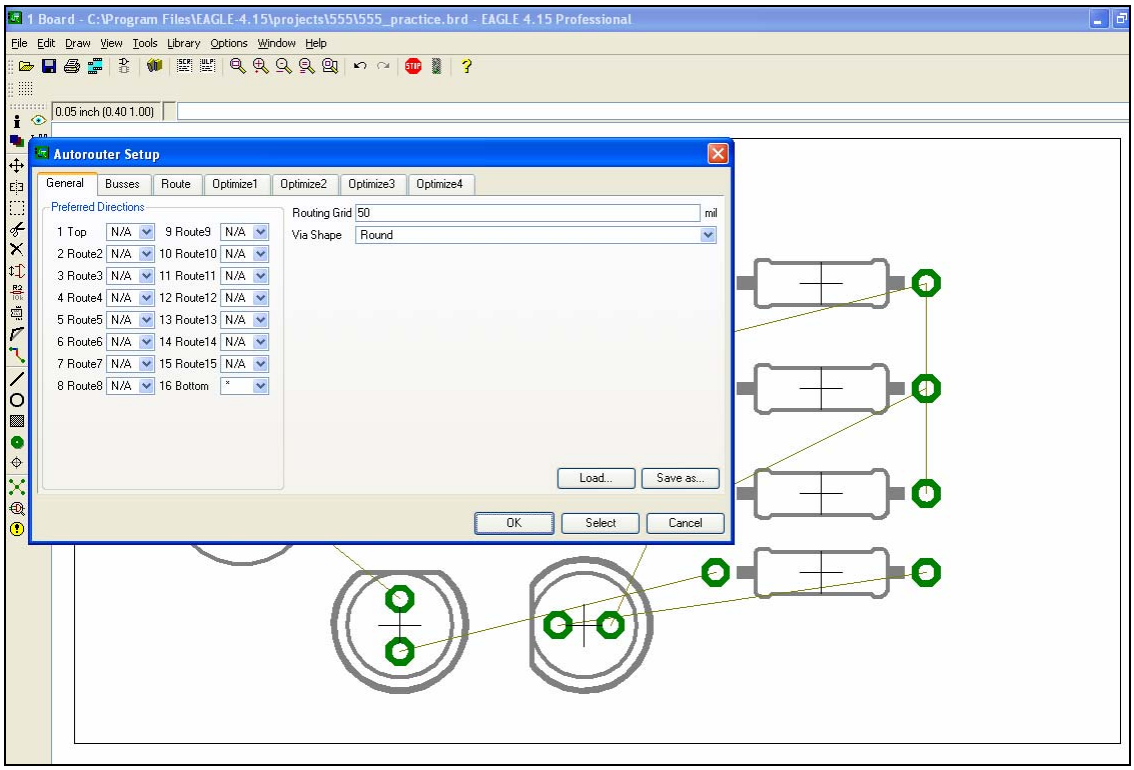
Autorouter

The Eagle Autorouter can route a board completely if the user has set the necessary parameters for the routing strategy and has placed components in a logical manner.

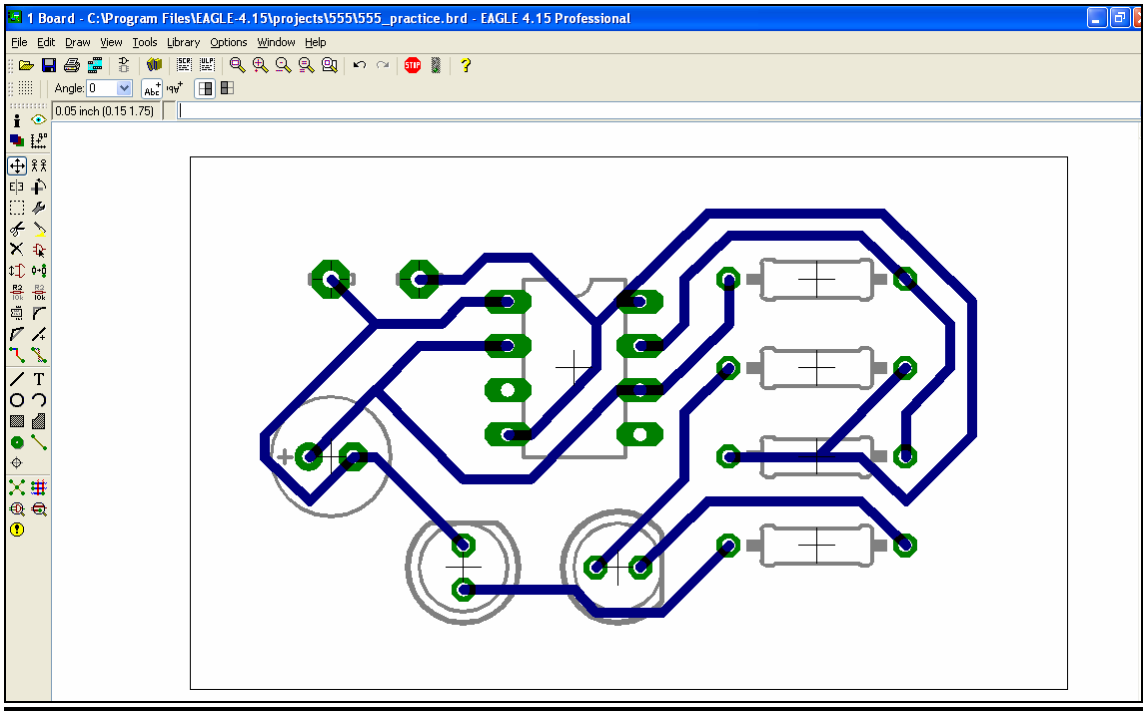
To run Autorouter:

- select Tools>Auto or click on the Auto toolbar icon command.

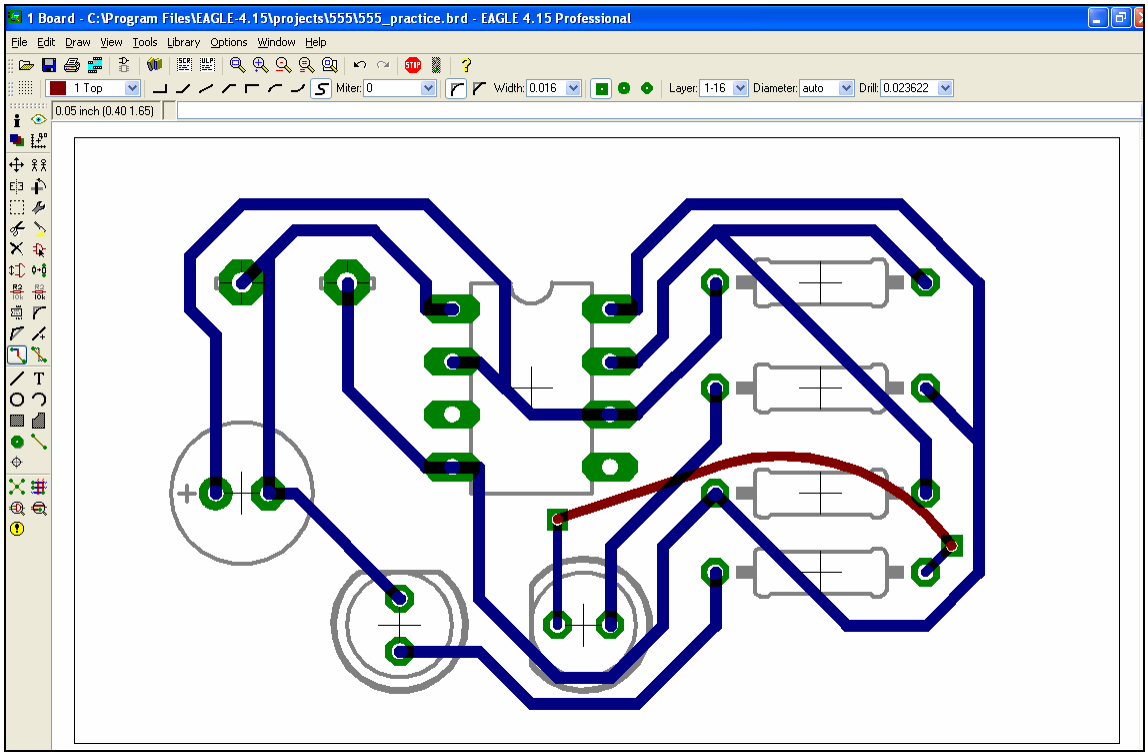
The screen shot below displays the Autorouter Setup box that pops up after clicking on the Auto command.


1 Board - C:\Program Files\EAGLE-4.15\projects\555\555_practice.brd - EAGLE 4.15 Professional
File Edit Draw View Tools Library Options Window Help
0.05 inch (0.40 1.00)
Autorouter Setup
General Busses Route Optimize1 Optimize2 Optimize3 Optimize4
Preferred Directions
1 Top N/A 9 Route9 N/A
2 Route2 N/A 10 Route10 N/A
3 Route3 N/A 11 Route11 N/A
4 Route4 N/A 12 Route12 N/A
5 Route5 N/A 13 Route13 N/A
6 Route6 N/A 14 Route14 N/A
7 Route7 N/A 15 Route15 N/A
8 Route8 N/A 16 Bottom *
Routing Grid 50 mil
Via Shape Round
Load... Save as...
OK Select Cancel

The General tab displays routing directions for applicable board layers. The star in 16 Bottom box indicates that the tracks will autoroute with all possible bends or angles.
Autorouter routed all board traces in the layout shown below.


1 Board - C:\Program Files\EAGLE-4.15\projects\555\555_practice.brd - EAGLE 4.15 Professional
File Edit Draw View Tools Library Options Window Help
Angle: 0
0.05 inch (0.15 1.75)

Pictured below is a board layout that did not route completely using Autorouter. A manually routed jumper wire, shown as a curved trace, makes the circuit connection were Autorouter failed.



To make jumper connections on the component side of the board as displayed above (Top layer is displayed red in Layout Editor):

1. select Draw>Via or click the Via command icon
2. place the vias for the unconnected components in the desired locations on the board
3. select Edti>Change>Layer or the Change icon command>Layer
4. select Top
5. route the trace between vias.

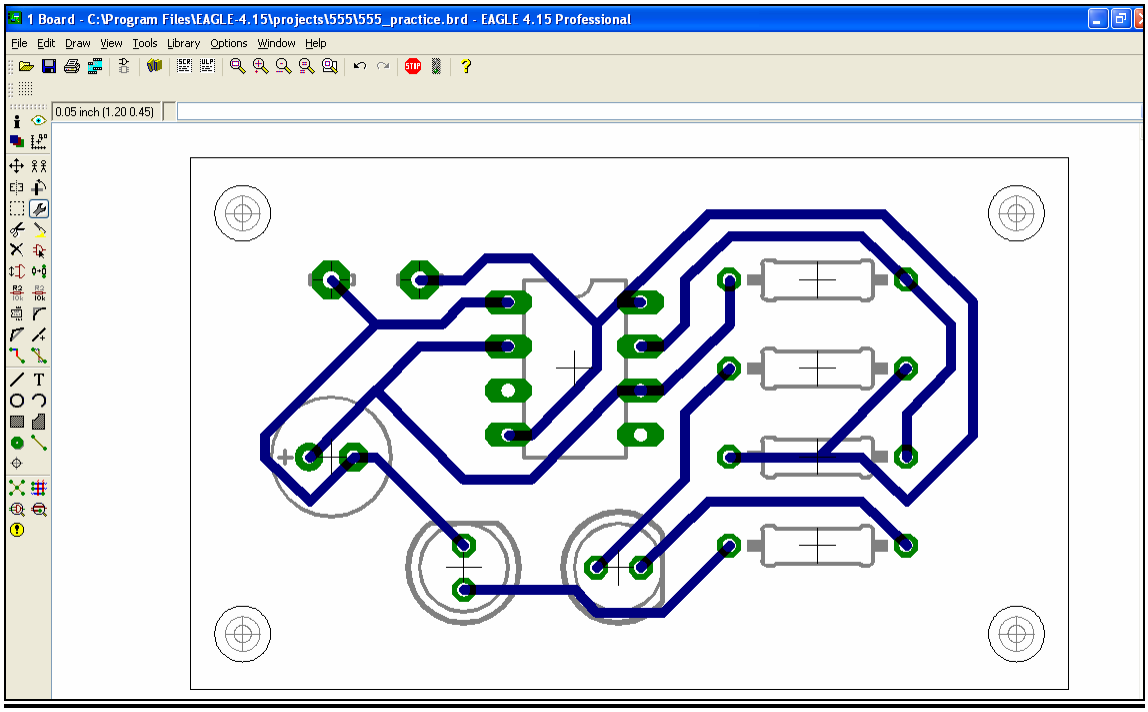
To add mounting holes for a board:

1. select Draw>Hole or click on the Hole command icon
2. place the holes in desired locations.

To change the size of a drill bit needed for a mounting hole:

1. Edit> Change>Drill
2. choose the size of drill bit needed.

The circuit board below has 1/8” (.125) drill bit holes placed near the corners for mounting.


1 Board - C:\Program Files\EAGLE-4.15\projects\555\555_practice.brd - EAGLE 4.15 Professional
File Edit Draw View Tools Library Options Window Help
0.05 inch (1.20 0.45)

Preparing Board Layout Files for IsoPro Transalation

This section of the tutorial describes the process of preparing definition files for board drill and track specifications. These files import into the IsoPro software, which controls the T-Tech routing machine. Before beginning, save the board under the project name.

Creating Drill files

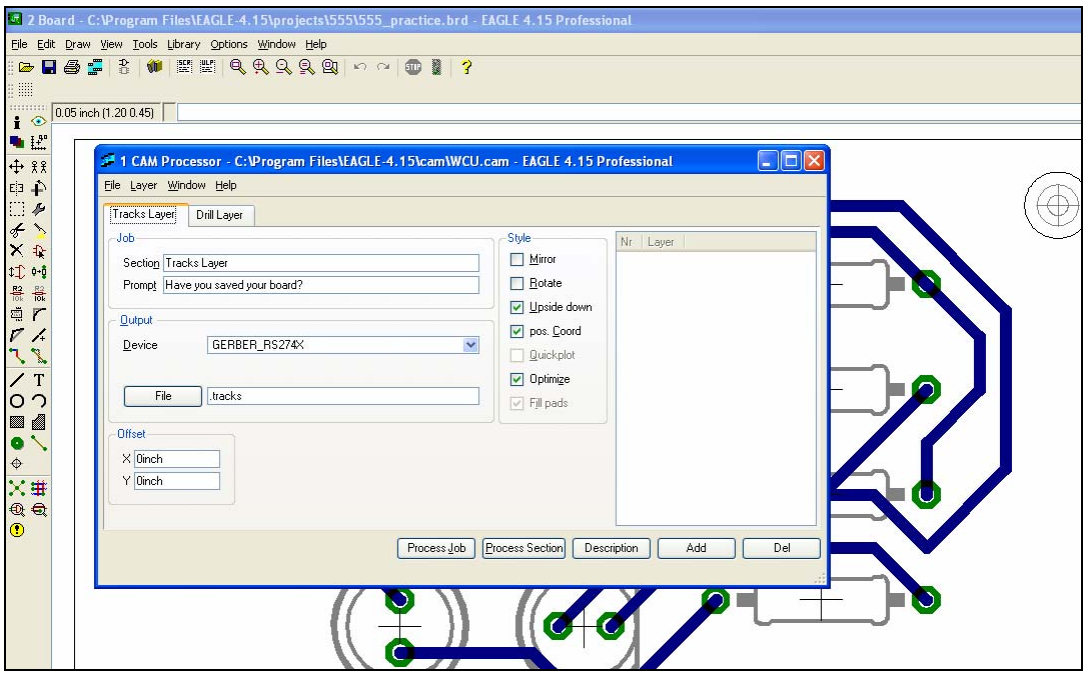
From the board window:

1. Select File>Run
2. Select the file drillcfg.ulp (script command: run drillcfg)
3. In the Eagle Drill Configuration box, check inches for output file units
4. click OK
5. click OK for the drill bit sizes
6. Save the configuration file under the project

Cam Processor

From the board window:
1. select File>Open Job
2. select WCU.cam

The CAM Processor will open up displaying the tracks layer with GERBER_RS274X selected as the Output Device.

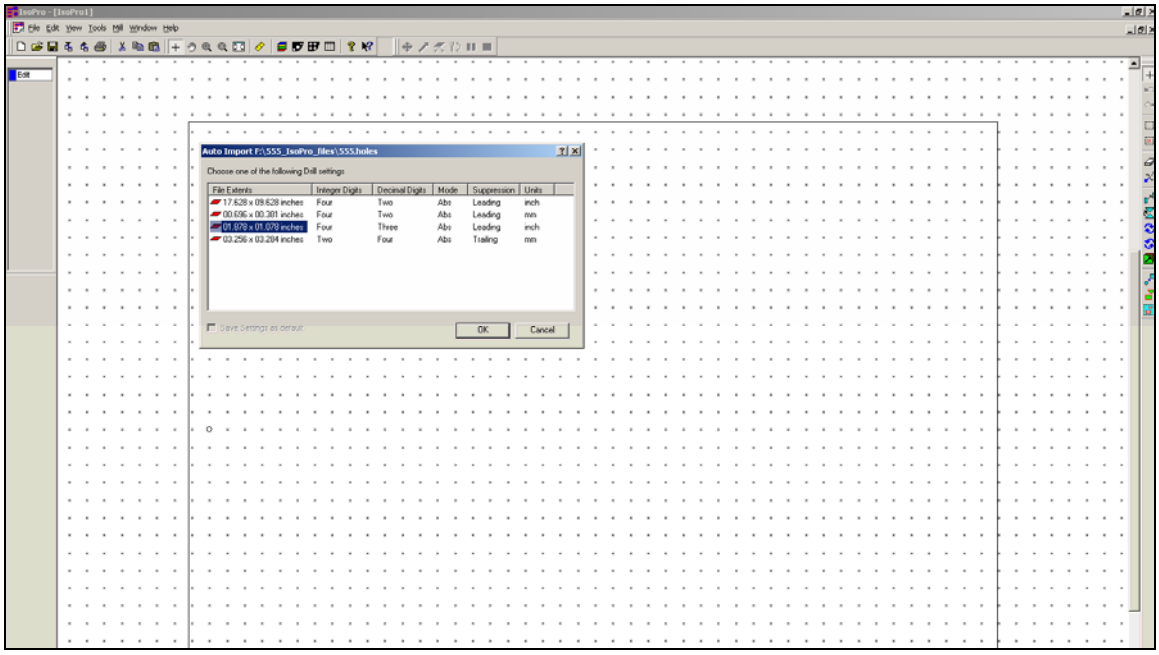


From the CAM Processor window
1. Select File>Open>Board
2. Select the .brd (board) file to be processed
3. Select Process Job
4. When prompted “Have you saved your board”, click OK
5. When prompted “Do NOT save changes to this job”, click OK

Save the two board files, .holes and .tracks, to some type of media storage device so that they can be imported into the IsoPro program.

Importing Gerber Files into IsoPro

1. Open IsoPro
2. click File>Import>Auto Detect Files
3. Open the .holes file from the directory where your board files are located.
4. press enter, a dialog box appears.
5. Highlight the drill setting with leading suppression and units in inches whose dimensions most closely match the board outline.
6. Click OK.



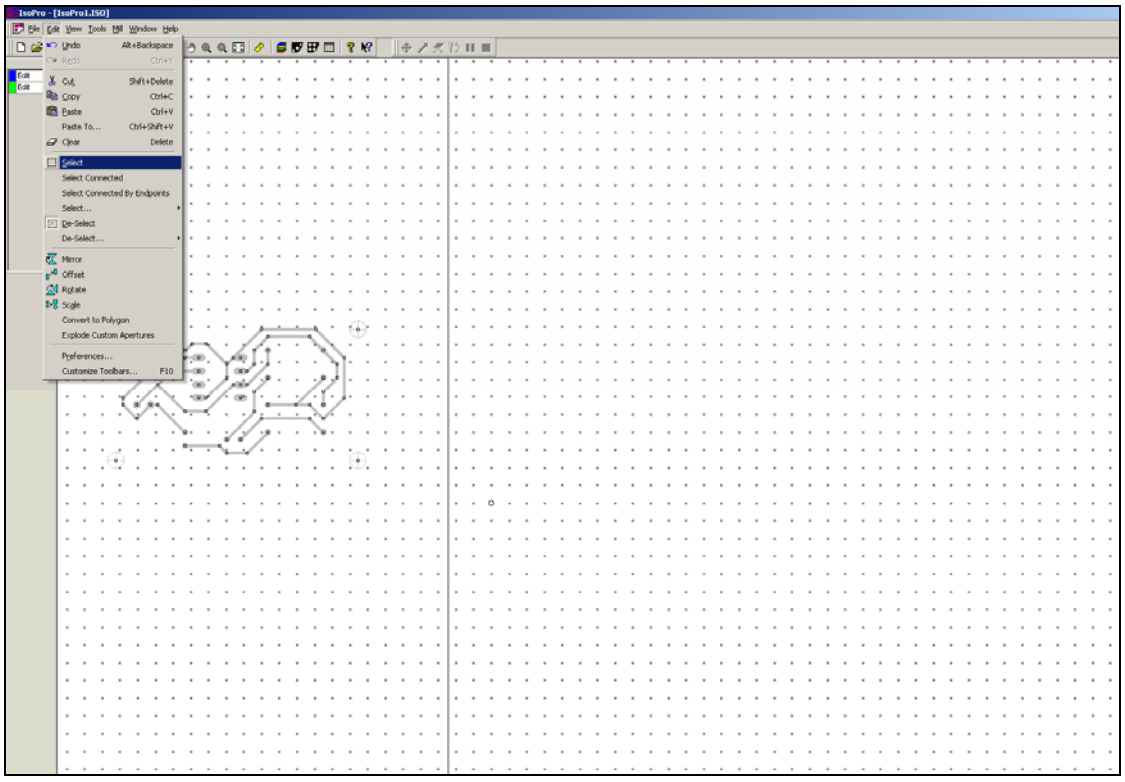
Repeat the import process for the .tracks file. Choose the file with leading suppression that most closely matches the dimensions of the .holes file that was imported.

Moving the Pattern Onto the IsoPro Board Area

1. click View>Zoom Out to view the entire drill and track pattern
2. click Edit>Select
3. drag a box around the pattern (the pattern is now grayed out)
4. change to the crosshair cursor

5. click and drag to move the selected pattern onto the outlined board area.

The screen shot below displays the grayed out drill hole and track pattern.

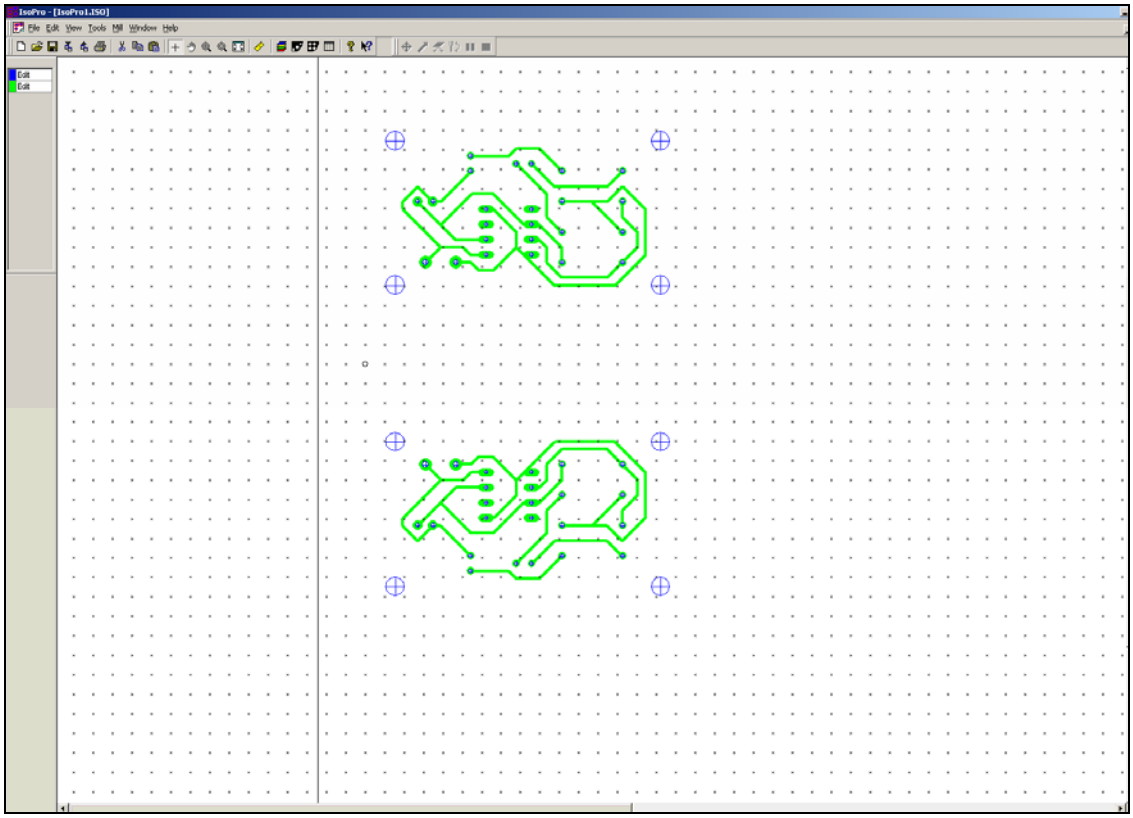


Mirroring the Board

Next, mirror the pattern so that the bottom or copper side will be facing upward for the routing of the circuit.

To mirror the pattern:
1. select Edit>Select
2. draw a box around pattern
3. click Edit>Mirror

For comparison, the screen shot below shows the mirrored image near the top of the screen with the original pattern below.

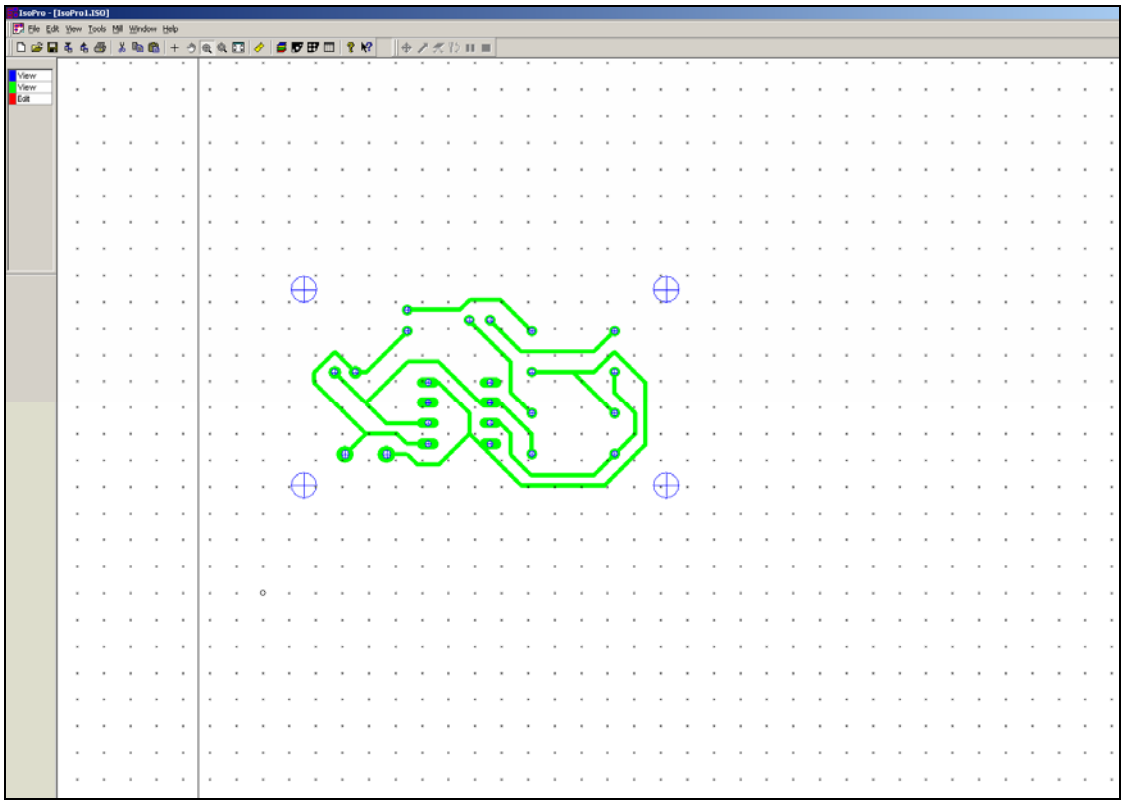


Border and Text Layers

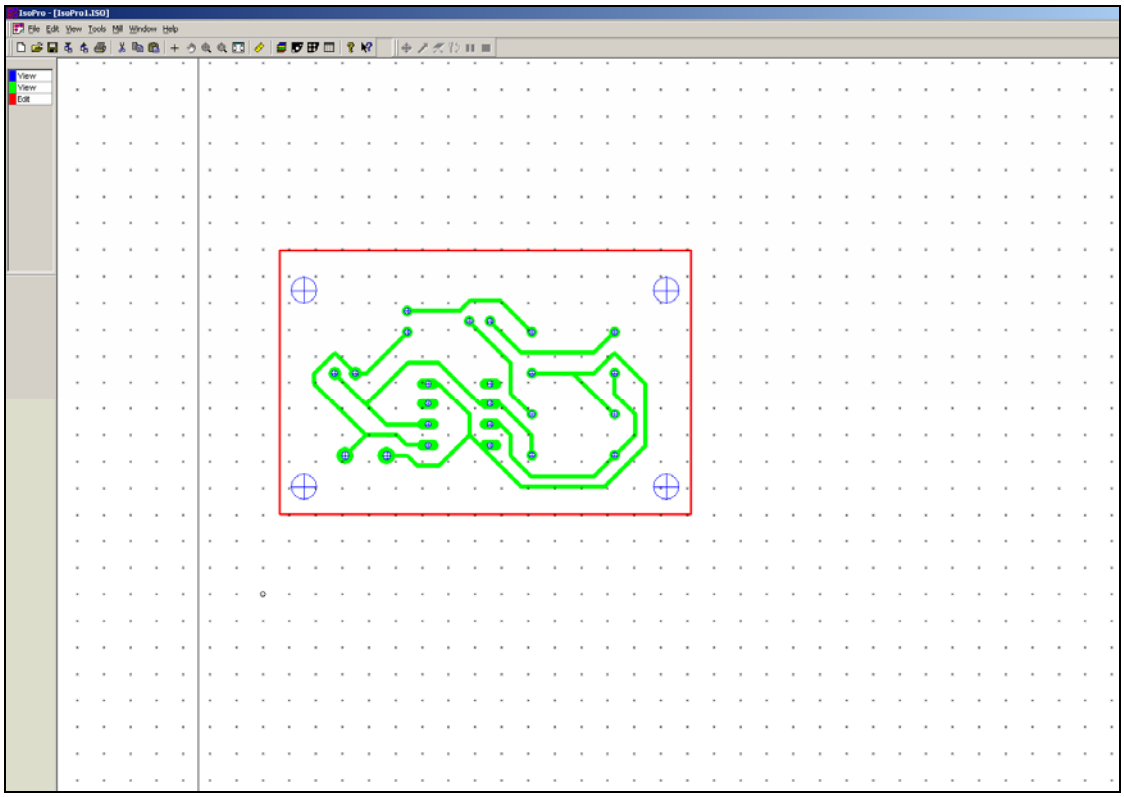
Create an outline of the board is on a new layer so that the circuit is separated from the rest of the copper plated board material when the routing is complete.

To create a new layer:

1. Select View>Layer Table
2. click on new layer
3. Click each layer icon in the upper left corner of the screen to set all but the new layer on “View” as shown in the next screen shot.



1. click on Tools>Create Rectangles
2. Draw a rectangle around the circuit as shown below:



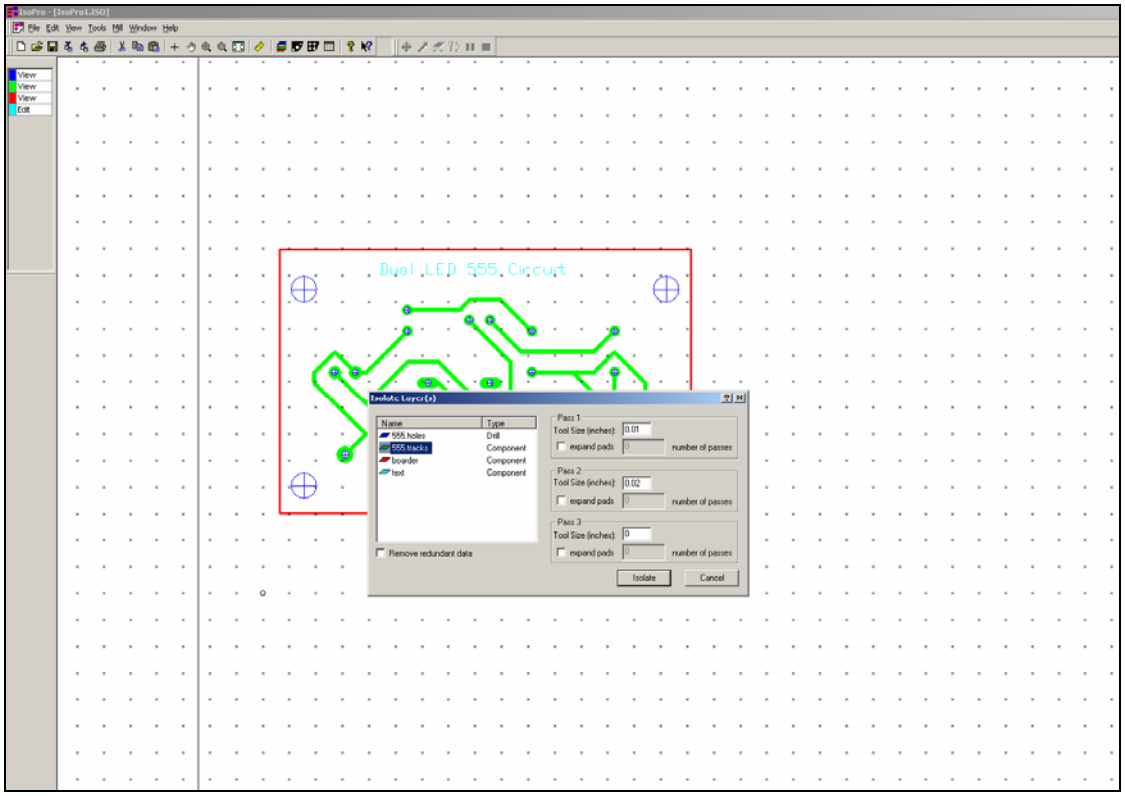
Text may be added to the design, if desired.
To add text:

1. View>Layer Table
2. click on new layer.
3. Click each layer icon in the upper left corner of the screen and set all but the new layer on “View”.
4. select Tools>Create Text
5. enter the desired text in the text box
6. set the text size to .05 (inches) with vector font selected
7. place the text in an appropriate location on the board

Track Isolations

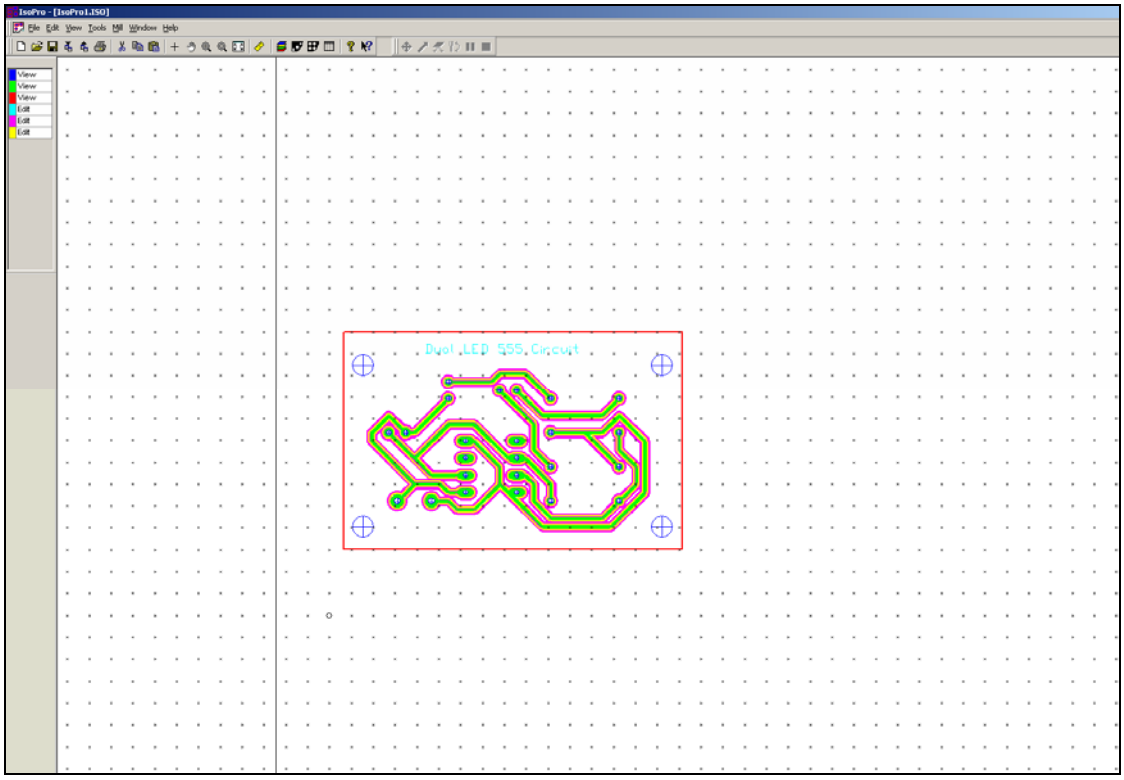
Isolations create an outline around the pads and tracks of the design so that the T-Tech routing machine can route the board. This process typically uses two isolations. IsoPro will produce a single isolation for tracks too close for two passes in.

1. Click Tools>Isolate, and observe the dialog box as shown below.



2. Select the layer that needs to be isolated
3. Choosing the appropriate isolation widths and passes. The default settings of .010” and .020” should be sufficient for hand-soldered boards.
4. click Isolate

The screen shot below displays the circuit tacks with 10 and 20 mil isolations.



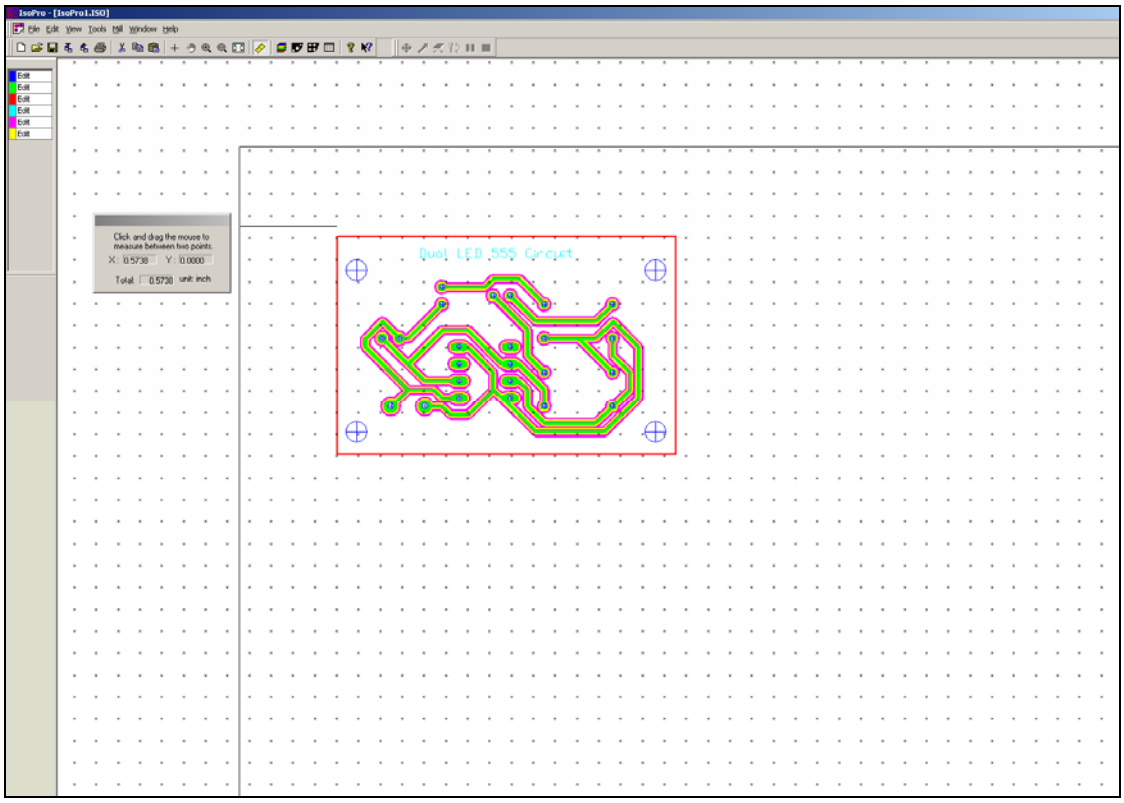
Board Placement

It is very important for proper board placement before beginning the routing process. The small circle midway between the top and bottom board edges and close (3 X’s over) to the left edge represents the actual dowel pin holding the copper plate in place on the Quick Circuit routing machine.
Note: the Quick Circuit router has built in limit switches that prevent routing within .5 inches from the edge of the copper plate.

To position the board for routing:
1. Find an open location on the copper board plate

2. Measure, from the closest edges of the copper plate, the X and Y position of the circuit border (see **Note** above).
3. Change all layers to edit
4. Select the circuit board and move it close to the area that corresponds to the copper plate measurements taken in step 2.
5. Select Tool>Measure
6. Use the measuring tool to determine the actual X and Y distances of the board's placement in relation to the borders.

The screen shot below displays the measuring tool's X dimension from the circuit border to that of the border representing the copper plate.

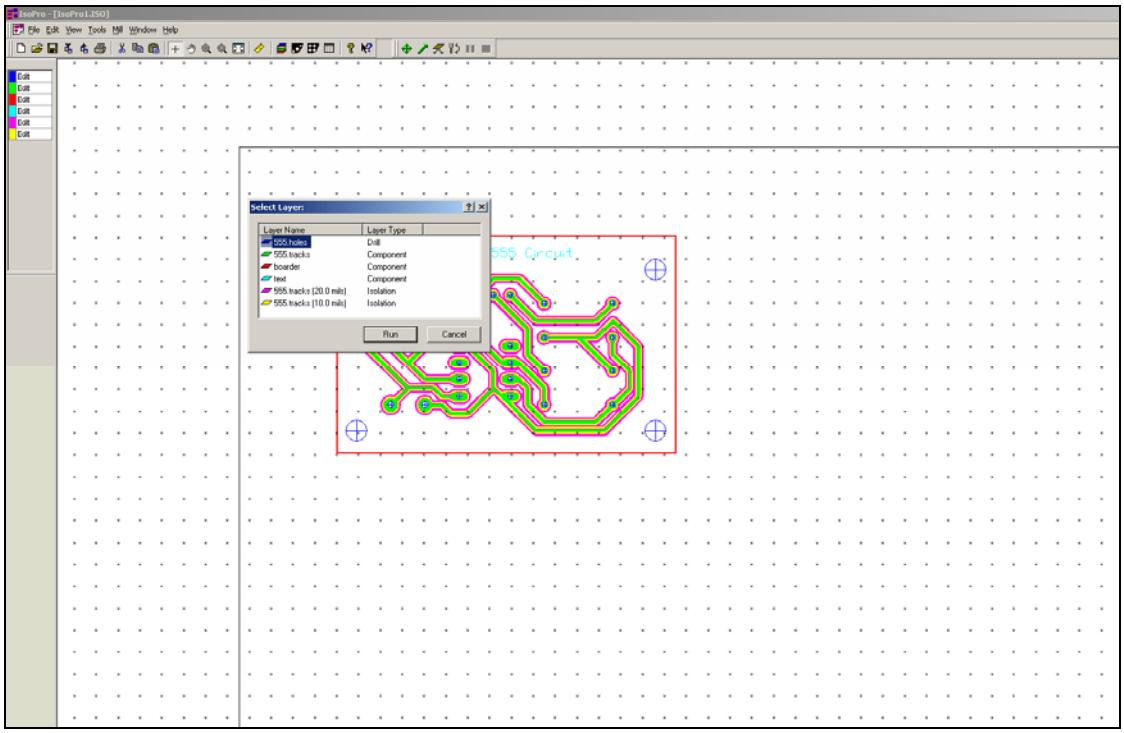


Routing the board:

Graphical descriptions of the tools used in routing the board are displayed on page 26 of the Quick Circuit User's Manual.

1. Click on Mill>Initialize to prepare the machine for routing the board.
2. Click Mill>Run Layer
3. select the drill layer

Drilling the holes first is important, since drilling them after the small pads have been cut may remove the copper from the board material. The Run Layer dialog box is shown below.



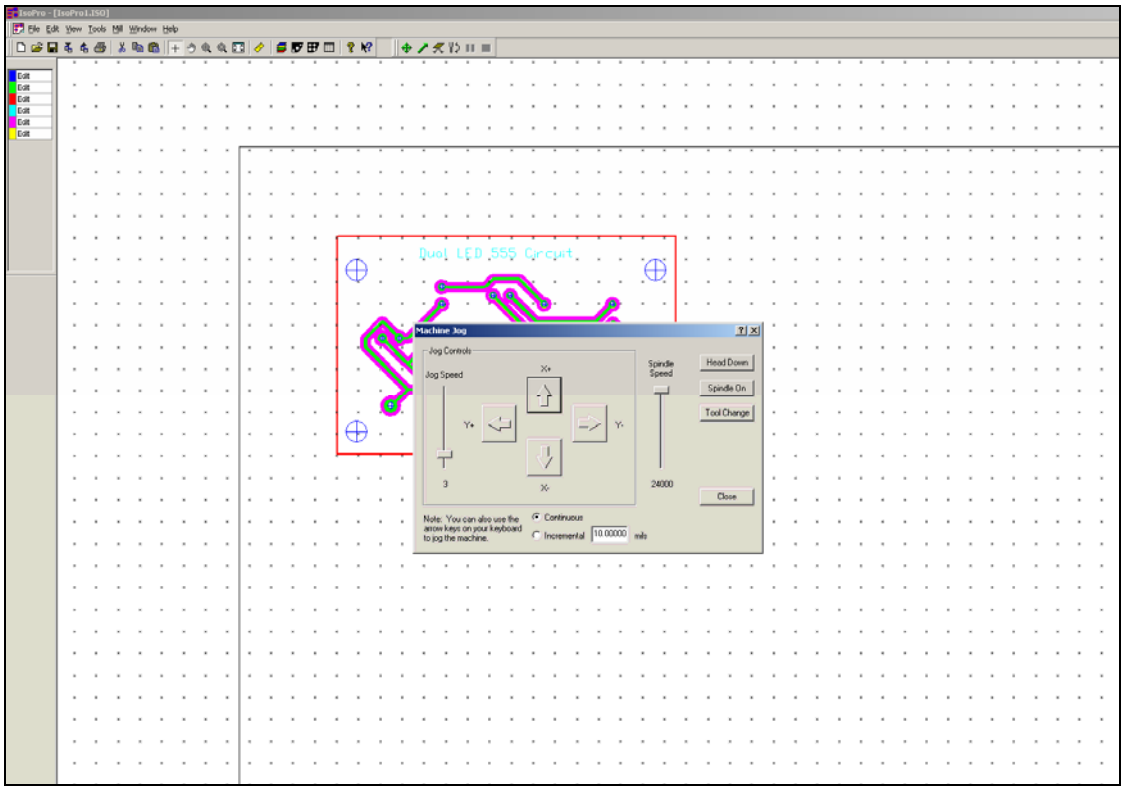
4. Select “Run” and insert the proper drill bit. A .032” drill is fine for most through-hole components and connectors.

5. Check for appropriate depth and execute the drilling cycle. If the circuit board has mounting holes, you will be prompted for a larger diameter drill bit.

6. Once the holes are complete, select the .010 layer. When prompted for tool change, replace the drill bit with the pointed tool for the 10 mil isolation. Tool depth determines the width of the cut. To check tool depth:
    a. select Mill>Jog
    b. move the milling tool to the area next to the Y-axis border with room enough to make a practice cut
    c. adjust the jog speed to the slowest position
    d. click spindle on
    e. click head down

f. clicking the Y-axis arrow, move the milling tool while adjusting the depth adjustment (+ or -) on the solenoid of the Quick Circuit router. + makes the cut deeper, - makes it shallower. It may be easier to use the keyboard arrows for this function.
g. After milling a cut of at least an inch
    i. click spindle off
    ii. click head up
h. select Mill>Move
i. when prompted, type 10 in the box. This moves the mill .010”
j. Repeat steps d and e
k. click the Y-axis arrow or keyboard arrow and move the mill in the opposite direction-<u>do not</u> adjust the depth
l. using the magnifying glass, check to see if there is a thin copper line between the two cuts just made. If not, repeat the process with a shallower cut.

The screen shot below displays the Machine Jog tool.



7. The 20 mil isolation layer routes using the end mil tool. Depth adjustment is not necessary with the milling of this layer.

a. select Mill>Run Layer
b. click on the .020 isolation
c. select Run
d. When prompted, make the end mil tool change.

8. If a text layer was included, use the same milling tool as that used for the 20 mil isolation layer.
    a. Select Mill>Run Layer
    b. Click on the text layer
    c. select Run
    d. use the same tool as in Step 6

9. The border will be the last layer routed.
    a. select Mill>Run Layer
    b. click on the border layer
    c. Insert the contour router when the Tool Change prompt appears.
    d. Carefully adjust the contour routing tool with the + or – depth adjustment so that the tool cuts through the board material.

Remove the routed circuit board. Any excess dust is removed by disconnecting the vacuum hose from the Quick Circuit routing machine.